INTERNATIONAL JOURNAL
FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

www.ijraset.com

Call: ○08813907089 | E-mail ID: ijraset@gmail.com

Validation of Standard K-Emodel for the Analysis of Annular Diffuser

Nagendra K. Sharma[1], Ashish Dixit[2]

[1,2]*Amity University Madhya Pradesh, Gwalior*

Abstract: A diffuser is a device for converting the kinetic energy of an incoming fluid into pressure. As the flow proceeds through the diffuser there is continuous retardation of the flow resulting in conversion of kinetic energy into pressure energy. Such a process is termed as diffusion. Diffuser forms an important part in flow machinery and structures. The present study involves the CFD analysis of adiffuser to evaluate the proximity of various models. The diffuser considered in the present case has both the hub and casings are diverging with unequal angles and hub angle keeping constant as 5°. The geometries of all the diffusers are calculated for constant area ratio 2 and cone angle 10°, 15°, 20°, 25°. The characteristic quantity such as pressure recovery coefficient distribution at hub and casing walls at various sections .Numerical predictions, made using Computational Fluid Dynamics (CFD), are presenteddescribing the behavior of the performance of conical diffuser with turbulence model i.e.Renormalization-group, STDk-εmodel.STDk-ε model is a new model to be used in this type of application that needs to be compared with wellestablishedmodels. The grid for the conical type geometry of the diffuser was created usingthe GAMBIT package, and the predictions from the CFD models have been obtained.

I. INTRODUCTION

Diffusers are used in engines to decelerate the entering air flow. The amount of deceleration and hence the performance of a diffuser is primarily dictated by the divergence angle, shown in Figure 1. For a fixed inlet width, a larger divergence angle will result in better performance, usually defined by a higher pressure gradient along the length of a diffuser. The divergence angle is limited by the advent of stall, flow separation along the walls of the diffuser. The separated region acts as a blockage and narrows the effective width of the diffuser. As a result, stall leads to decreasing pressure recovery and performance.
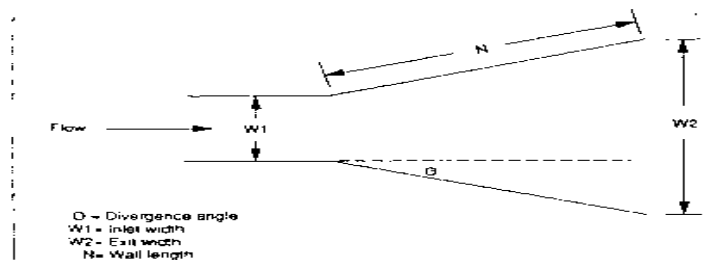

Figure 1: Diffuser Configuration

Diffusers are extensively used in centrifugal compressors, axial flow compressors, ram jets, combustion chambers, inlet portions of jet engines etc. The energy transfer in these turbo machineries involves the exchange of significant levels of kinetic energy in order to accomplish the intended purpose. As a consequence, very large levels of residual kinetic energy frequently accompany the work input and work extraction processes, sometime as much as 50% of the total energy transferred. A small change in pressure recovery can increases the efficiency significantly. Therefore diffusers are absolutely essential for good turbo machinery performance. The pressure recovery coefficient of a diffuser is most frequently defined as the static pressure rise through the diffuser divided by the inlet dynamic-head.

$$C_p = \frac{p_2 - p_1}{\frac{1}{2}\rho v_{av1}^2}$$

Where subscripts 1 and 2 refers to diffuser inlet and outlet conditions respectively.

v_{av1} represents the average velocity at the inlet.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

II.   MATHEMATICAL FORMULATION

Flow and related phenomenon can be described by partial differentiation or integro differential equations, which cannot be solved analytically except in special cases. To obtain an approximate solution numerically, one needs to use discretization method, which approximates the differential equations by a system of algebraic equations. The approximations are applied to small domains in space and/or time so that the numerical solutions provide results at distant locations in space and/or time. Much of the accuracy of the experimental data depends on the quality of tools used; the accuracy of the numerical solutions is dependent on the quality of discretizations used and the size and type of mesh. The mesh size and discretizations needs to be optimized for the convergence of the results.

A.   Turbulent Models

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix with transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. FLUENT provides the various choices of turbulence models named as Spalart-Allmaras model, k-ε models, Standard k-ε model , Renormalization-group (RNG) k-ε model ,Realizable k-ε model, Shear-stress transport (SST) k-ω model etc.STD k-ε   model is a new model to be used in this type of application that needs to be compared with well established models. The grid for the conical type geometry of the diffuser was created usingthe GAMBIT package, and the predictions from the CFD models have been obtainedusing the commercial CFD code FLUENT version 6.1STD k- ε □is well established model in predicting turbulent flows (Swirling) which have been compared successfully to experimental results.

B.   Governing Equations

The governing equations for 2D axisymmetric geometries are written as follows:
The continuity equation is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial r}(\rho v_r) + \frac{\rho v_r}{r} = S_m \quad (1)$$

Where x is the axial coordinate, r is the radial coordinate.
Conservation of momentum in an inertial (non-accelerating) reference frame is described by [1]

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla.(\rho \vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (2)$$

Where p is the static pressure,$\bar{\bar{\tau}}$ is the stress tensor (described below), and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces respectively. The stress tensor $\bar{\bar{\tau}}$ is given by:

$$\bar{\bar{\tau}} = \mu\left[\left(\nabla \vec{v} + (\nabla \vec{v})^T - \frac{2}{3}\nabla.\vec{v}I\right)\right] \quad (3)$$

Where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation

III.   CFD ANALYSIS

FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also refine or coarsen grid based on the flow solution.

IV.   SOLUTION PROCEDURE

The present study involves the CFD analysis of effect of swirl on flow characteristics. The annular diffuser considered in the present

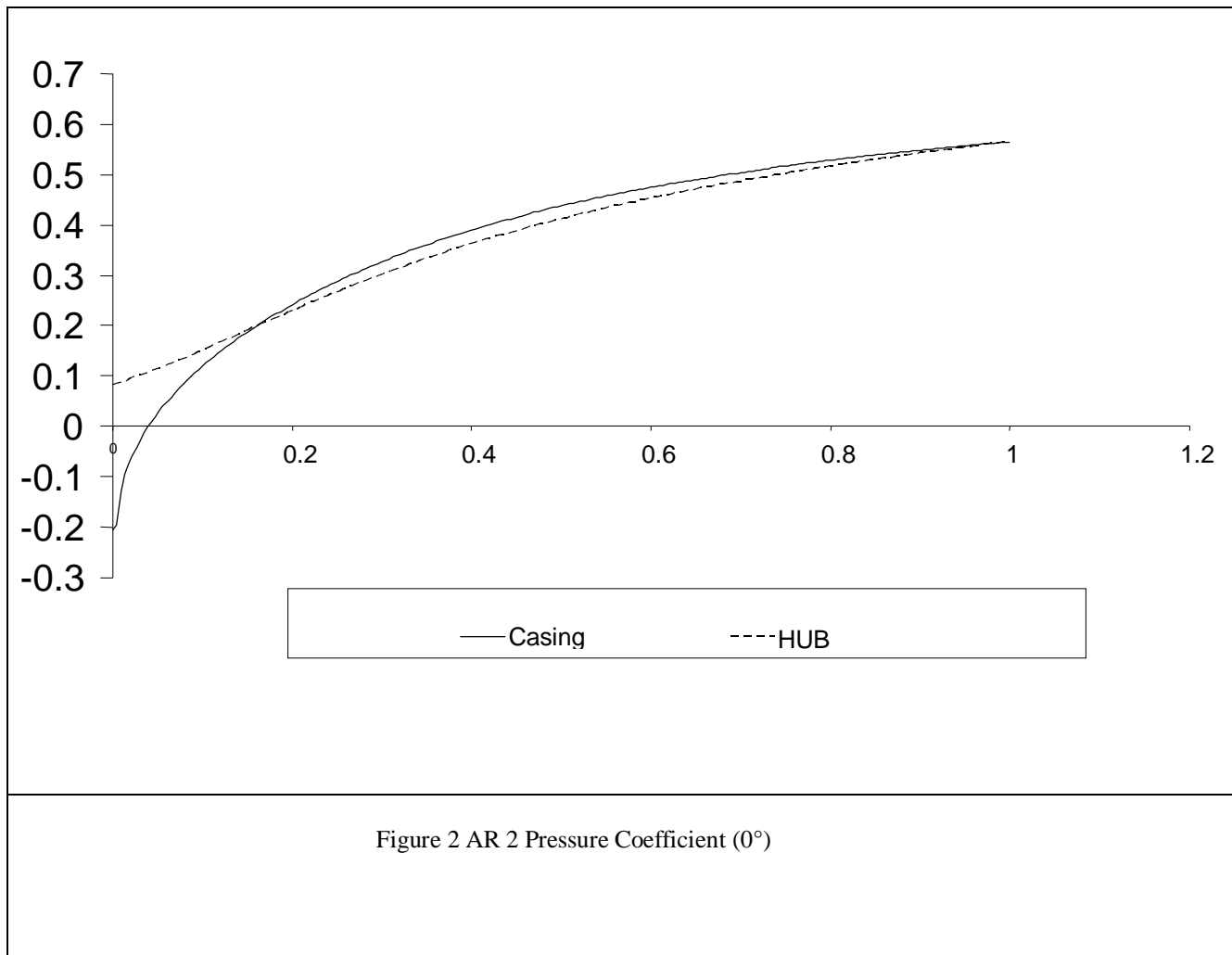# International Journal for Research in Applied Science & Engineering Technology (IJRASET)

case has both the hub and casings are diverging with unequal angles and hub angle keeping constant as 0°. The geometries of all the diffusers are calculated for constant area ratio 2 and angle 30. The geometry was created in GAMBIT and analysis was done in Fluent. The steps taken in GAMBIT are as below:

Step 1: Create Geometry

Step 2: Mesh Geometry in GAMBIT

Step 3: Specify Boundary Types in GAMBIT

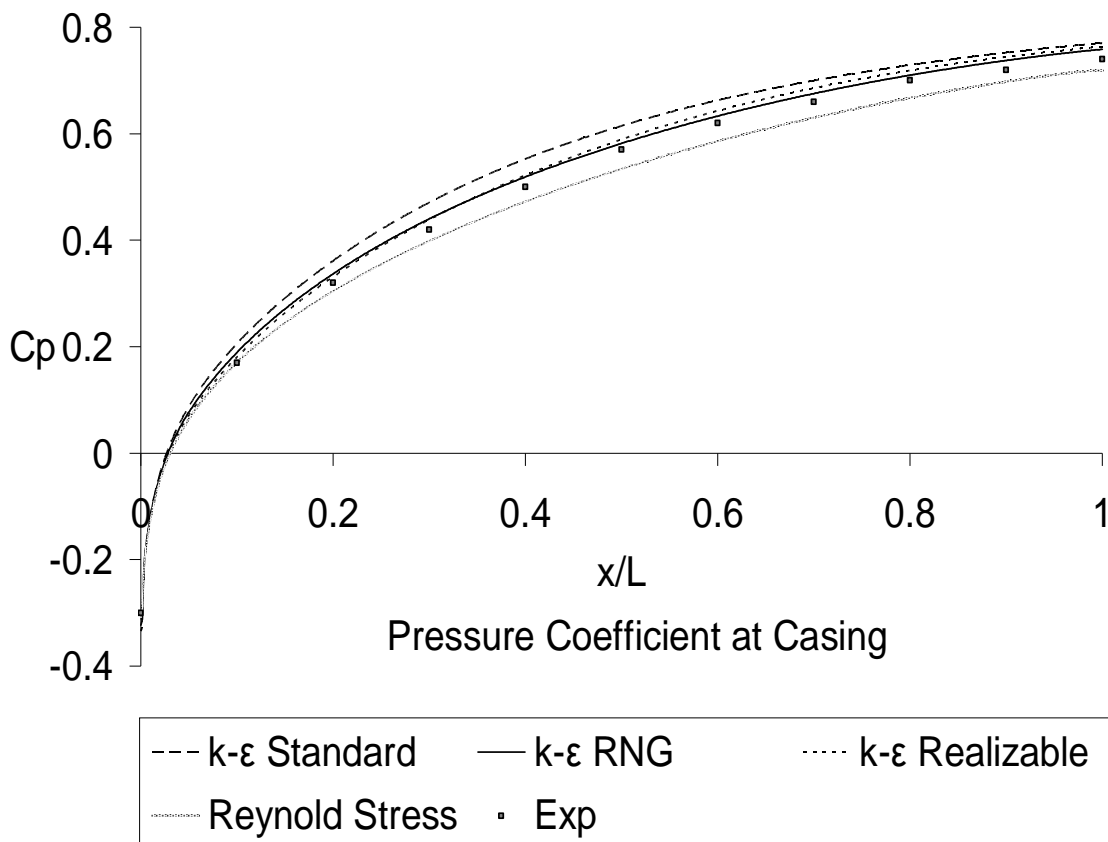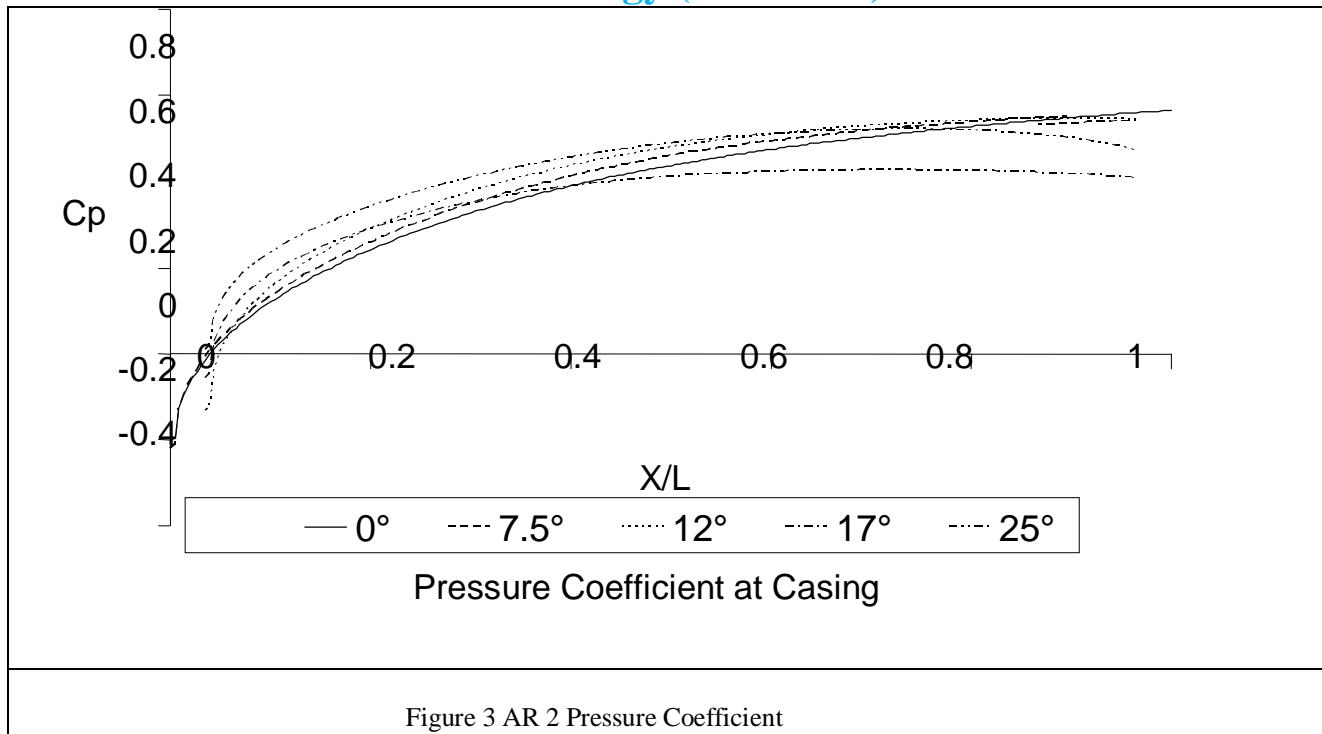## V.   MODELING INCOMPRESSIBLE FLOW IN A CONICAL DIFFUSER

A.   Copy the mesh file, diffuser. Mesh to your working directory.

B.   Start the 2D double precision (2ddp) solver of FLUENT.

IV. RESULT AND DISCUSSION

Figure 2 AR 2 Pressure Coefficient (0°)

9

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Figure 3 AR 2 Pressure Coefficient

Figure 4 AR 2 Pressure Coefficient

10

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

VI.   CONCLUSION

CFD analysis is reasonably in good agreement with the experimental data.

In Figure 4, the results obtained for Pressure recovery coefficient by using different models of turbulent, compared with experimental results. It is explicit from the graph the results obtained by using k-ε RNG model are closest to experimental values.

A.   Nomenclature

AR	Area ratio	y/Ym	Non-dimensional radial length
Cp	Pressure recovery coefficient	F	Force
I	Unit tensor	L	Diffuser length
P	Static pressure	Re	Reynolds number
V	Velocity in Y direction	α	Divergence angle
ρ	Density	μ	Molecular viscosity
$\bar{\bar{\tau}}$	Stress tensor	x/L	Non-dimensional axial length

B.   Suffix

x   axial

r   radial

z   tangential or swirl

REFERENCES

[1]   G. K.Batchelor,"An Introduction to Fluid Dynamics", Cambridge University Press, Cambridge, England, 1967

[2]   Howard, J.H.G., Henseler, H.J and Thorriton-Trurnp, A.B., "Performance and Flow Regimes of Annular Diffusers", ASME Paper No. 67-WA/FE-21, 1967.

[3]   Sovran, G and Klomp, E.D., "Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular, Conical or Annular Cross-Section", Fluid Mechanics of Internal Flow, Ed. G. Sovran, Elsevier Amsterdam, pp.270, 1967.

[4]   Srinath.T, "An investigation of the effects of swirl on flow regimes and performances of annular diffuser with inner and outer cone angles." Thesis, University of waterloo, Canada 1968.

[5]   Ackert, J. 1967. Aspect of Internal Flow. Fluid Mechanics Of Internal Flow ,Ed. Sovaran G., Elsevier Amsterdum,pp1.

[6]   Adkins R.C, JacobsenOH , Chevealier P 1983 A Premilary Study of Annular Diffuser With Constant Diameter Outer Wall. ASME paper no. 83-GT-218

[7]   Adkins R.C., 1983. A simple Method For Design Optimum Annular Diffusers. ASME Paper No. 83-GT-42.

[8]   Arora, B.B., Pathak, B.D., 2005 "Flow characteristics of parallel hub diverging casing axial annular diffusers". ISME publication pp 794-798

[9]   Cockrell, D.J., Markland, E., 1963 .A Review of Incompressible Diffuser Flow. Aircraft Engg. Volume 35, pp 286.

[10]   Coladipietro, R., Schneider, J.M., Sridhar, K.1974. "Effects of Inlet Flow Conditions on the Performance of Equiangular Annular Diffusers," Trans. CSME 3 (2): pp. 75-82.

[11]   Dovzhik, S.A., Kartavenko, V.M., 1975. "Measurement of the Effect of Flow Swirl on the Efficiency of Annular Ducts and exhaust Nozzles of Axial Turbo machines," Fluid Mechanics/Soviet Research 4(4): 156-172.

[12]   Goebel, J. H., Japikse, D., "The Performance of an Annular Diffuser Subject to Various Inlet Blockage and Rotor Discharge Effects," Consortium Final Report, Creare TN-325, March 1981.

[13]   Hesterman R, Kim S, Ban Khalid A, Wittigs 1995. Flow Field and Performances Characteristics of Combustor Diffusers: A Basic Study. Trans. ASME Journal Engineering for Gas Turbine and Power 117: pp 686-694.

[14]   Hoadley D, 1970. Three Dimensional Turbulent Boundary Layers in an Annular Diffuser. PhD Thesis University of Cambridge.

[15]   Hoadley, D., Hughes, D.W., 1969. "Swirling Flow in an Annular Diffuser," University of Cambridge, Department of Engineering, Report CUED/A-Turbo/TR5.

[16]   Howard, J. H. G., Thorton –Trump A. B., Henseler H. J. 1967" Performance And Flow Regime For Annular Diffusers".ASME paper no . 67-WA/FE-21.

[17]   Ishkawa K, Nakamura I 1989"An Experimental Study on The Performance of Mixed Flow Type Conical Wall Annular Diffuser ASME FED-69.

[18]   Japikse, D., 1986. "A New Diffuser Mapping Technique – Studies in Component Performance: Part 1," ASME Paper No. 84-GT-237, Amsterdam, June 1984; also, Journal of Fluids Engineering, Vol. 108, No. 2. pp. 148-156.

[19]   Japikse, D., and Pampreen, R., "Annular Diffuser Performance for an Automotive Gas Turbine," ASME Publication 78-GT-147. 1978.

[20]   Japikse, D.,1980. "The Influence of Inlet Turbulence on Diffuser Performance," Concepts ETI, Inc., Design Data Sheet No. 1, .

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)